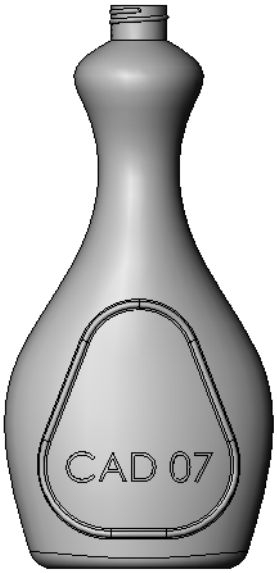
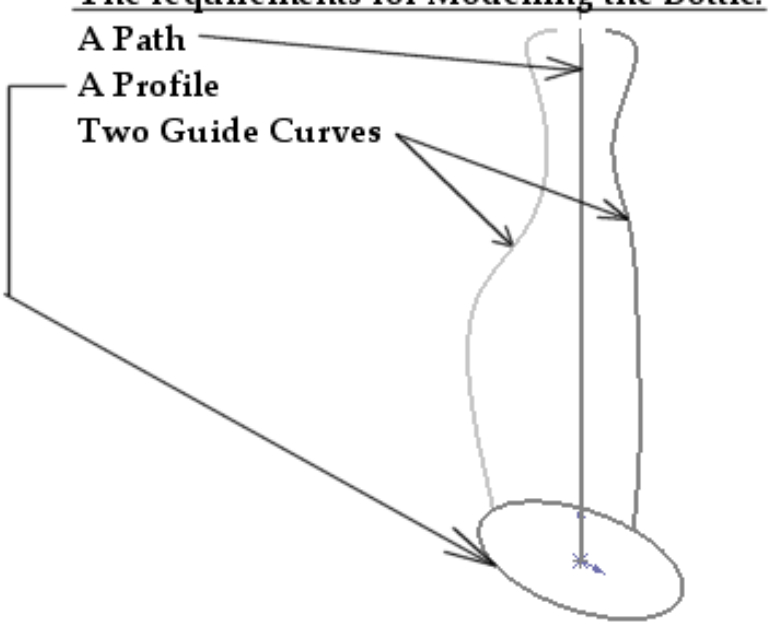
SolidWorks Workshop
Institute of FIeld roBOtics

Sweeping and Lofting

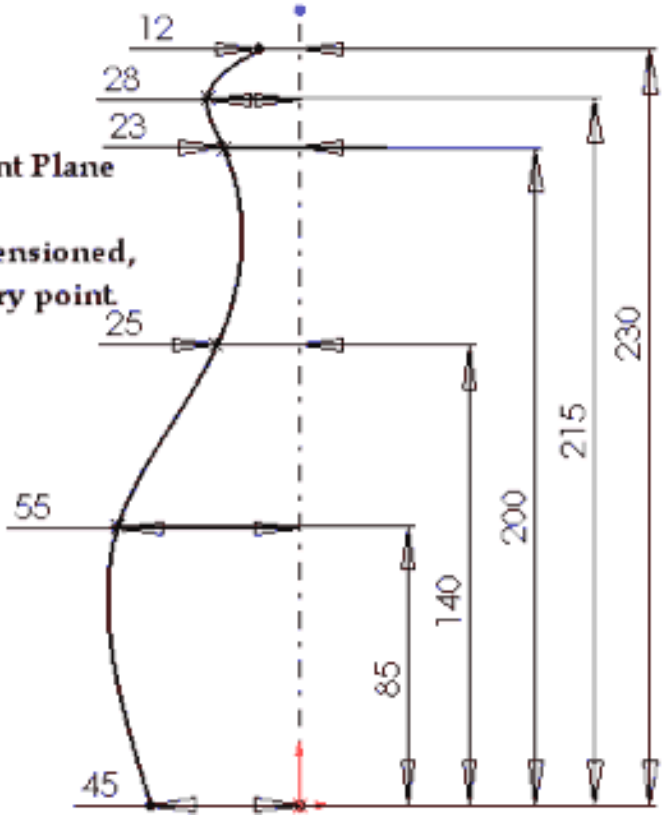
Exercise 1: Bottle



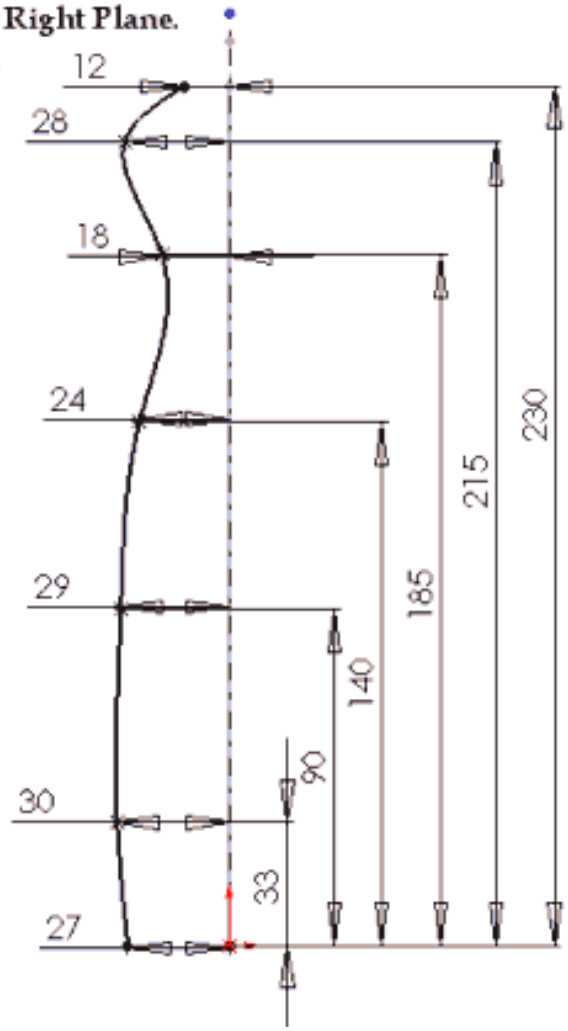
The requirements for Modelling the Bottle:



Draw the following Profile on the Front Plane
Use a Spline to create the curve firstly.
Each point on the spline must be dimensioned,
and will need two dimensions for every point.
Make sure to fully define the Spline.



Draw the following Profile on the Right Plane.
Use the same Method as previous.

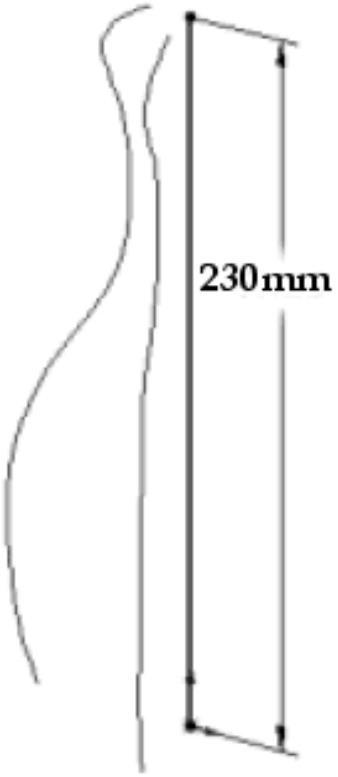


Sweep path.

Select the `Front` reference plane and open a sketch.

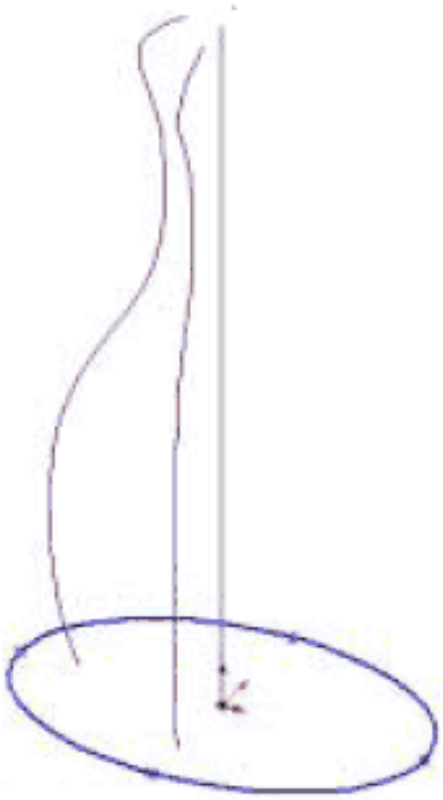
Sketch a vertical line, starting at the `Origin`.

This will be used as the sweep path.



Sweep section.
Select the `Top` reference plane and open a sketch.

On the Sketch Tools toolbar, click the **Ellipse** tool and sketch an ellipse with its center at the `Origin`.



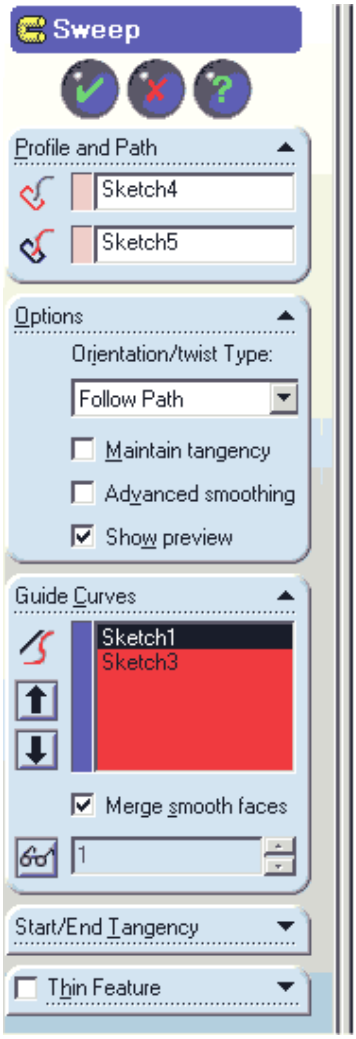
Relating the sweep section to the guide curves.
The profile of the sweep section has to be related to the guide curves using the **Pierce** relation. Thus, the guides had to be created *before* the profile.

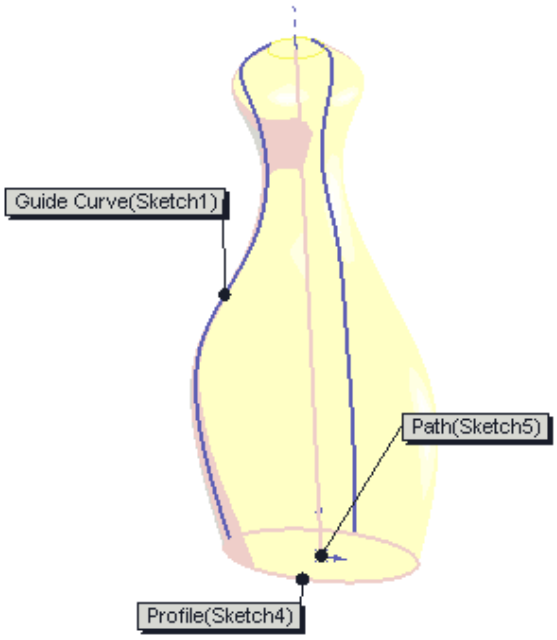
Press the **Ctrl** key, and select the point at the end of the major axis and the first guide curve. Right-click, and select **Make Pierce**. Repeat this procedure for the minor axis and the second guide curve.

Fully defined.
Since the **Pierce** relation on the major axis defines its size *and* orientation, we do not need to further constrain it. If we had used a dimension to control the size of the major axis, we would need to control the orientation of the major axis in some way

Exit the sketch.
The sweep section is now fully defined so you can exit the sketch. We are now ready to sweep the first feature.

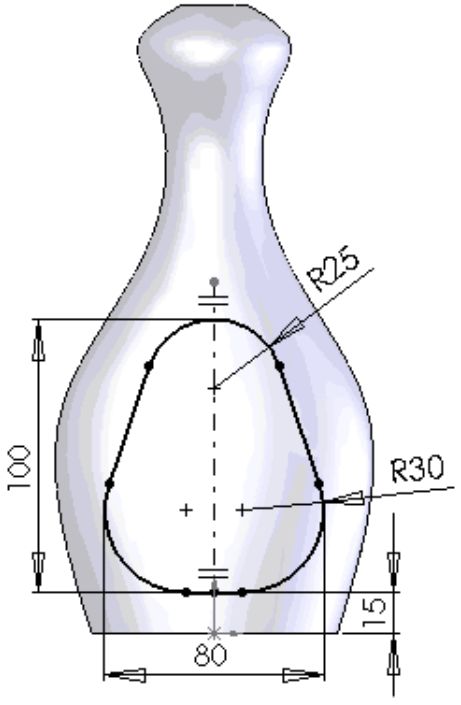
Unlike extruded or revolved features, swept features cannot be created while active in a sketch. You must exit the sketch first. This is because swept features require multiple sketches which you identify individually.





The Label Shape:

Draw the Following Profile on the Front Plane.



Introducing: Insert Projected Curve

Projected Curve projects a sketch onto a face or faces of the model. When these faces are curved, the result is a 3-dimensional curve.

Where to Find It

- Click  on the Curves toolbar.
- Or, click **Insert, Curve, Projected...**.

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21 Projected Curve dialog and preview.

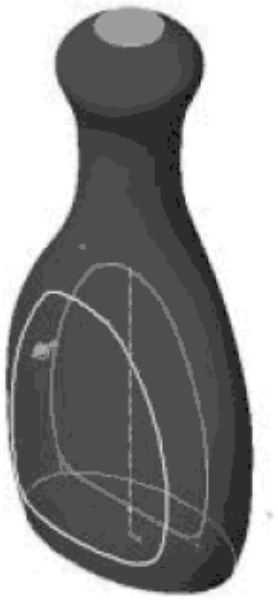
Click  , or on the **Insert** menu, click **Curve, Projected...**. Choose the **Sketch onto Face(s)** option from the list.

22 Selections.

Click in the **Sketch to Project** list and select the sketch. Click in the **Projection Faces** list and select the model face.

By default, the system projects the sketch normal to the sketch plane (along the positive Z axis). If you want to project the curve onto the back of the bottle, click **Reverse Projection**.

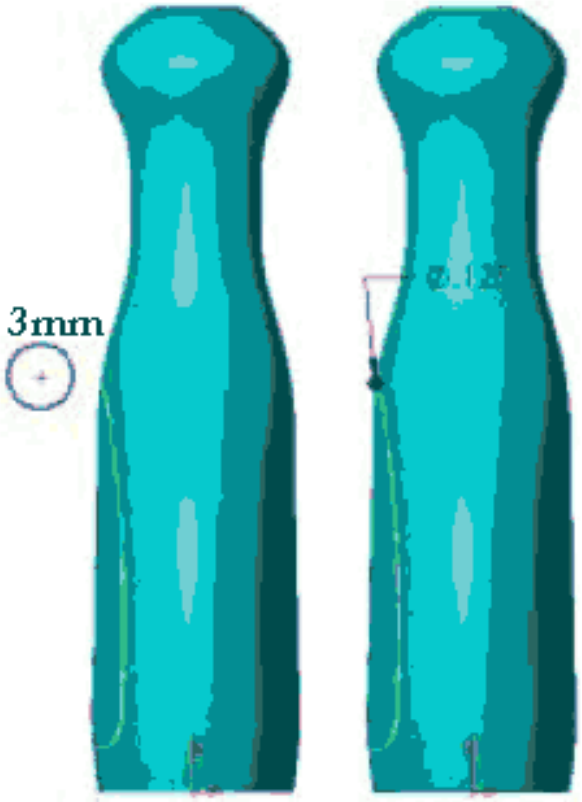
Click **OK**.



Projected curve.
The system projects the sketch onto the front surface of the bottle. This curve will be used as the sweep path to create a boss to outline the area the label area on the bottle.

Sketch the profile.
Change to a `Right` view and select the `Right` reference plane. Open a sketch and draw a circle in any convenient location.

Pierce relation.
Add a **Pierce** relation between the center of the circle and the projected curve to define its location. Dimension the circle to 3mm diameter.



The projected curve pierces the sketch plane in two places: at the top and the bottom. The system chooses the pierce point closest to where you select the curve. If you want the circle located at the top, select the projected curve near the top. It's that simple.

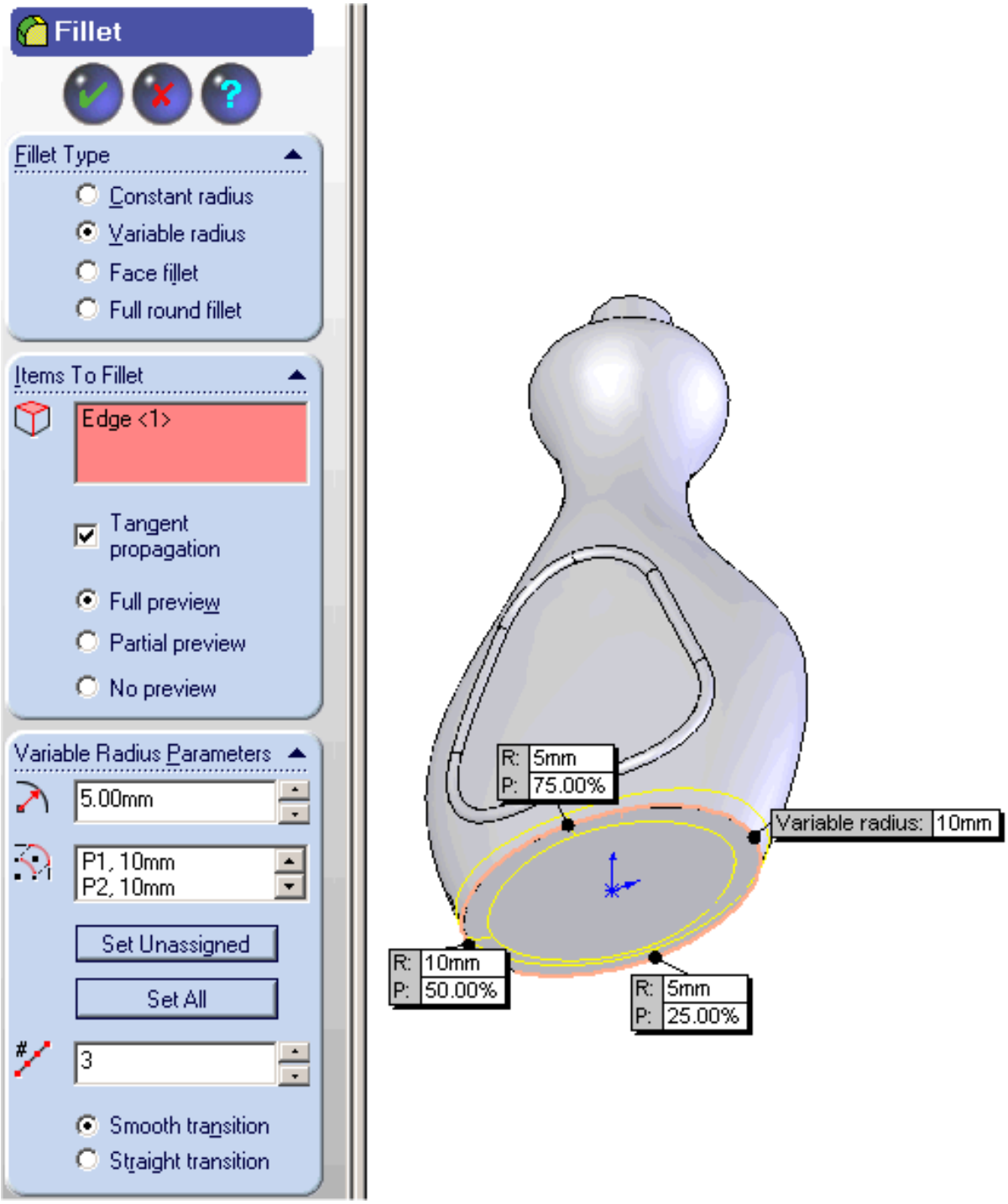
Sweep the boss for the label outline.
Exit the sketch.

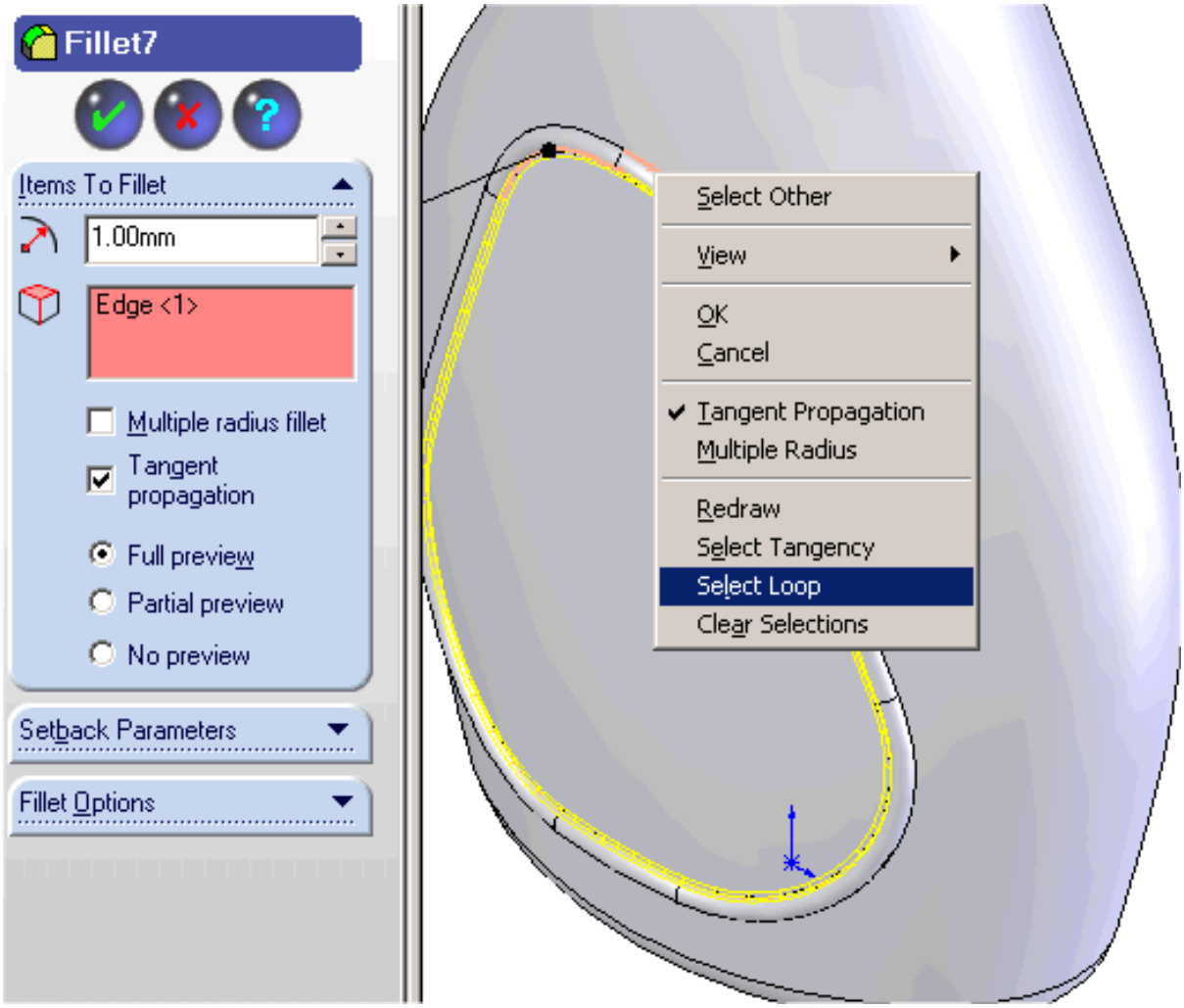
Click  . Select the circle as the **Profile** and the projected curve as the **Path**.

Click **OK**.

Notice the system has no difficulty sweeping a feature with the profile located at the middle of a closed path.

Add the neck.
Select the top face of the bottle and open a sketch. Use **Convert Entities**  to copy this edge into the active sketch. Extrude the sketch upward a distance of **15mm**


Fillet
Fillet Type
Constant radius
Variable radius
Face fillet
Full round fillet
Items To Fillet
Edge <1>
Tangent propagation
Full preview
Partial preview
No preview
Variable Radius Parameters
5.00mm
P1, 10mm
P2, 10mm
Set Unassigned
Set All
3
Smooth transition
Straight transition
R: 5mm
P: 75.00%
Variable radius: 10mm
R: 10mm
P: 50.00%
R: 5mm
P: 25.00%



Shell command.

Click  on the Features toolbar, or click **Insert, Features, Shell...**

Set the **Thickness** to 1mm as the default.

For the **Faces To Remove**, select the top face of the bottle neck.

Multiple thickness.

Expand the **Multi-thickness Settings** section. Face selections here will not be the default thickness.

Select thicker faces.
Click in the **Multi-thickness Faces** field and select the outside face of the bottle neck. Set the thickness to **2mm**

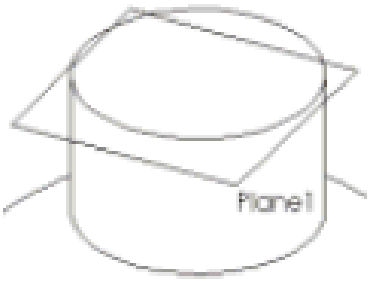
Click **OK** to create the shell.

Results shown in section view.
The illustration at the right shows a section view, viewed from the back.

Save your work.
We have invested a lot of time into this case study. Now would be a good time to save the file.

In the remainder of this example, we will build the threads on the neck of the bottle as shown at the right.

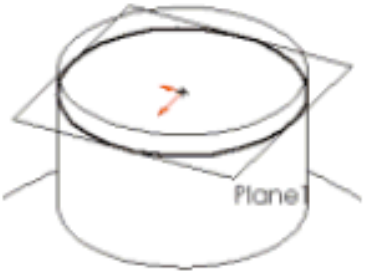
Offset plane.
Create a reference plane offset **2mm** *below* the top of the bottle neck. This is where the threads will start.



Insert sketch.
With this plane selected, open a new sketch.

Copy the edge.

Copy the edge of the bottle neck into the active sketch using **Convert Entities**  . This circle will determine the diameter of the helix.

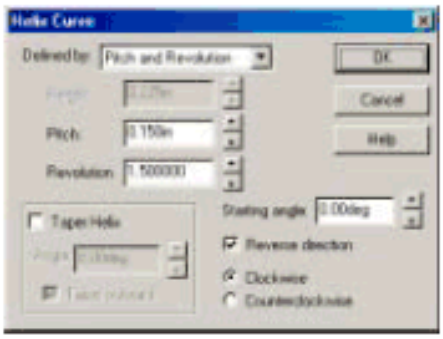


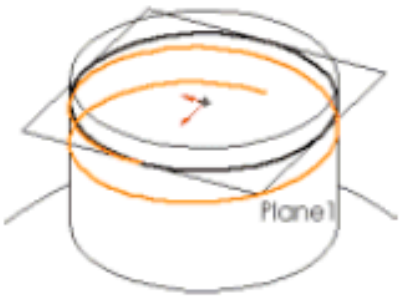
Create the helix.

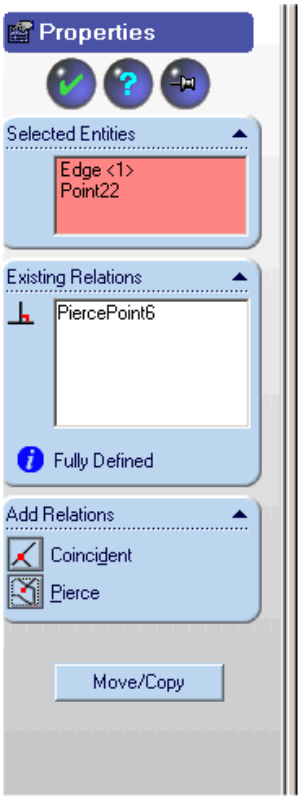
Click  . The **Helix Curve** dialog is used to specify the definition of the helix. The threads have a **Pitch** of **4mm** for **1.5 Revolutions**. The threads are **Clockwise** and go down the neck from a **Starting Angle** of **0°**.

As you change the parameters of the helix, the preview graphics update to show the result.

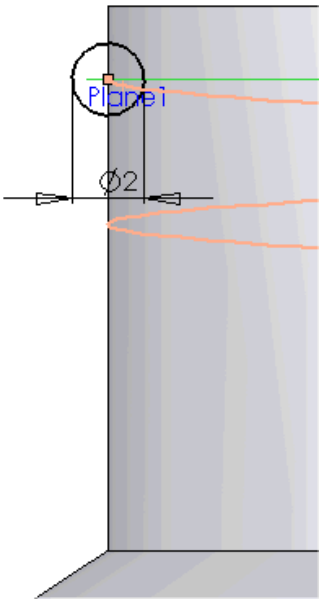
Click **OK** to create the helix.





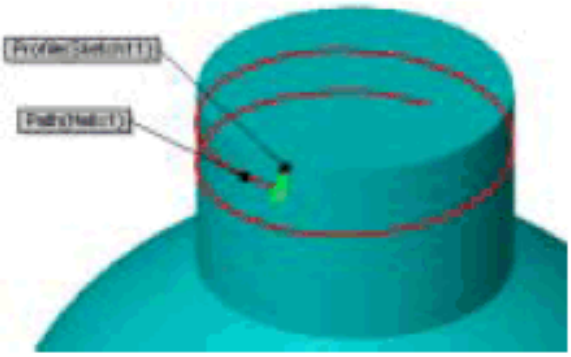


Insert a Sketch on the Right Plane as Shown.
Use a 'Pierce' Relation to define the profile sketch.



Exit the sketch.

Sweep the threads.



Click  . Select the sketch as the sweep section, and the helix as the sweep path.

Click **OK**.

If you are wondering what the option **Align with End Faces** is used for, we will cover a simple example explaining its purpose after we finish with the bottle. See *Align with End Faces* on page 125.

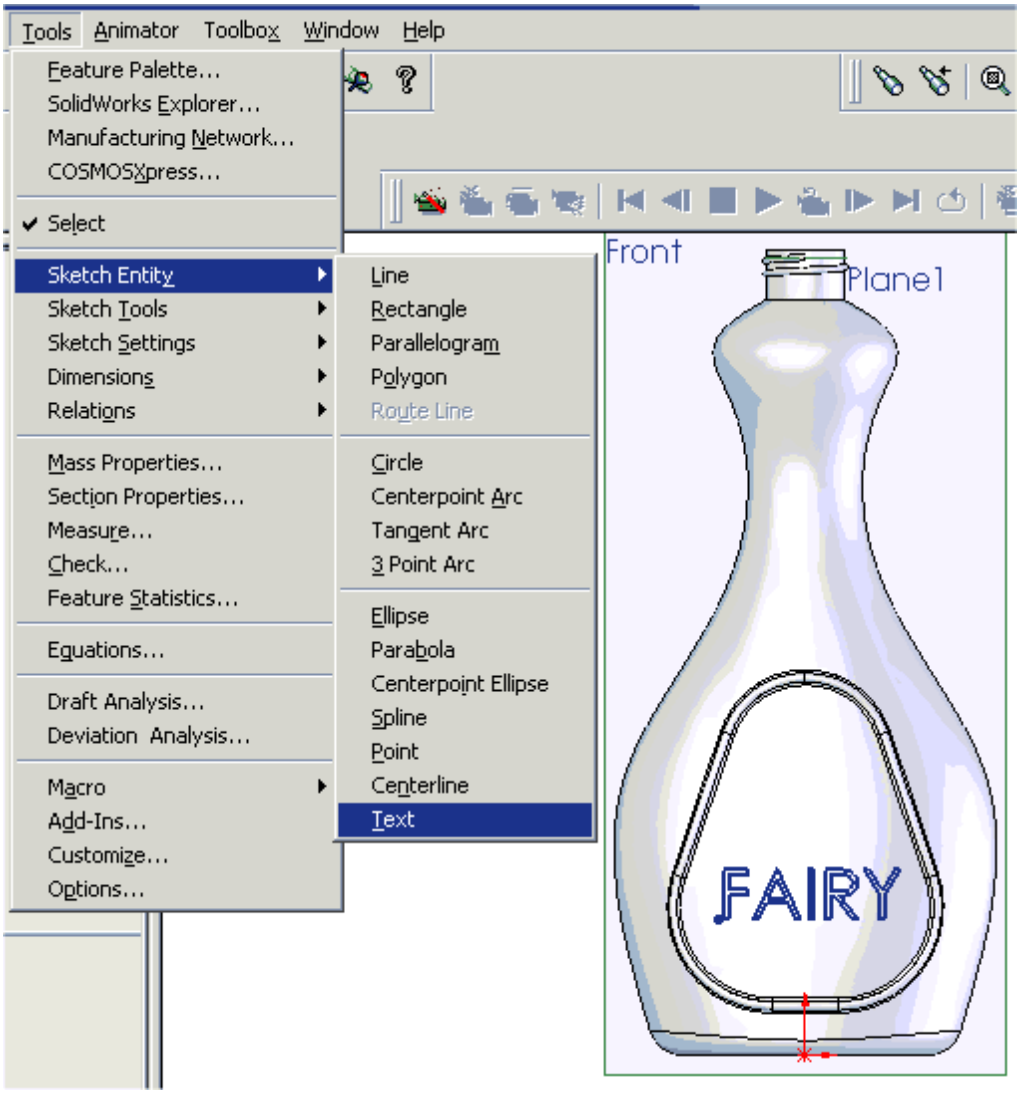
Results.

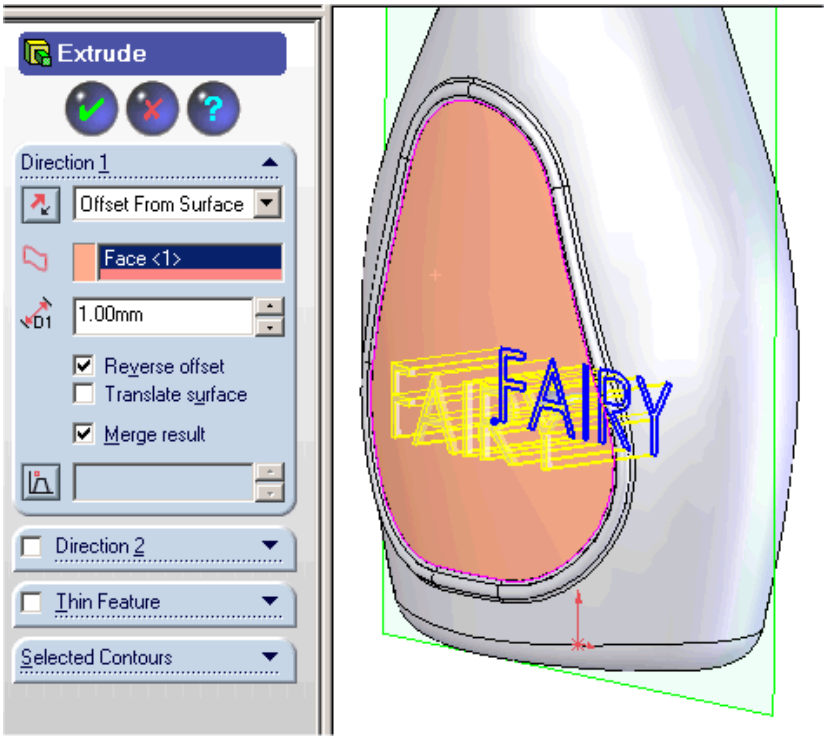
The results of sweeping the thread are shown at the right.

Add the finishing details.

An easy way to round off and finish the ends of the thread is to create a revolved feature. Do this for both ends of the thread.

An easy way to create the centerline that is needed for the revolved feature is to use **Convert Entities**  to copy the vertical edge where the thread meets the body of the neck. Then change the line's properties to **Construction Line** and you have your centerline.


Tools
Animator
Toolbox
Window
Help
Feature Palette...
SolidWorks Explorer...
Manufacturing Network...
COSMOSXpress...
Select
Sketch Entity
Sketch Tools
Sketch Settings
Dimensions
Relations
Mass Properties...
Section Properties...
Measure...
Check...
Feature Statistics...
Equations...
Draft Analysis...
Deviation Analysis...
Macro
Add-Ins...
Customize...
Options...
Line
Rectangle
Parallelogram
Polygon
Route Line
Circle
Centerpoint Arc
Tangent Arc
3 Point Arc
Ellipse
Parabola
Centerpoint Ellipse
Spline
Point
Centerline
Text
Front
Plane1
FAIRY


Extrude
Direction 1
Offset From Surface
Face <1>
1.00mm
Reverse offset
Translate surface
Merge result
Direction 2
Thin Feature
Selected Contours
FAIRY

Exercise 2: Funnel (Lofting and Sweeping)

Funnel

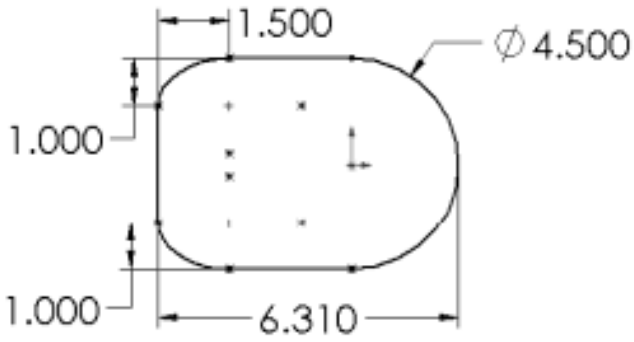
Create this part using the information and dimensions provided. This lab reinforces the following skills:

- Lofting
- Shelling
- Sweeping

Procedure

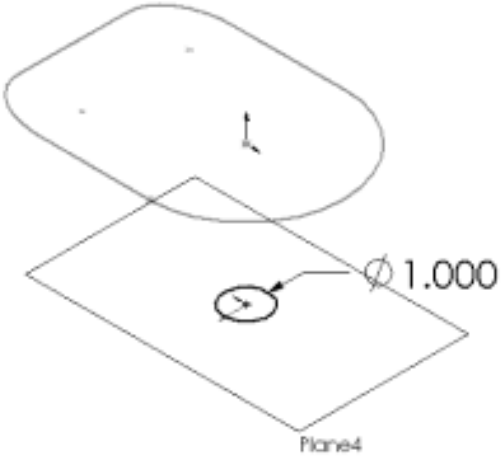
Open a new part using the `Part_IN` template and name it `Funnel`.

1 Sketch the first profile.
Use ellipses, lines and arcs to create this profile.

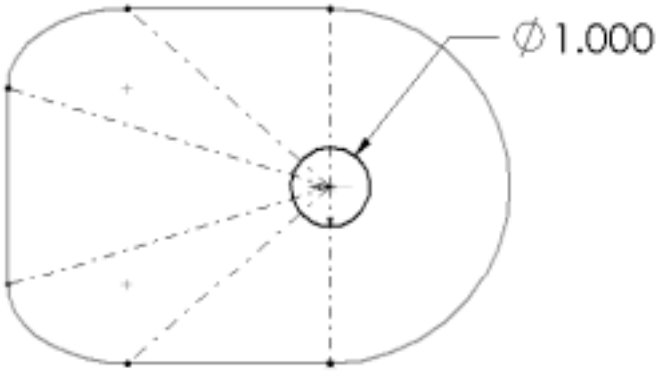


2 Second profile.
Create a new plane that is parallel to the Top reference plane **3.25"** below it. Sketch a circle lined up with the `Origin`.

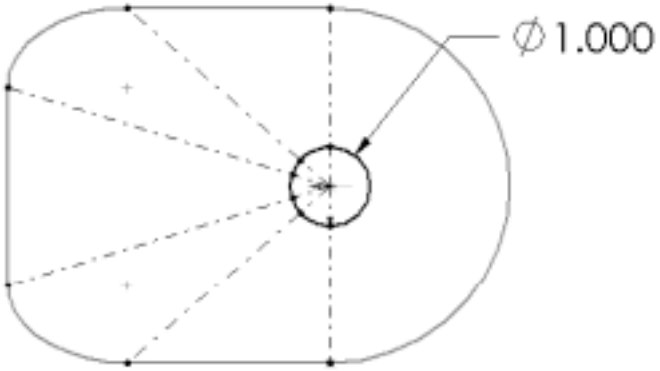
This circle will be used as the second profile in a loft, after it is divided up into sections that match the first profiles endpoints. If the circle is not broken up, the loft will decide what the breakup of the circle should be.



3 **Breakup.**
Add centerlines radially from the circle's center to the endpoints of the first profile. This geometry will cross the circle's circumference at several places.



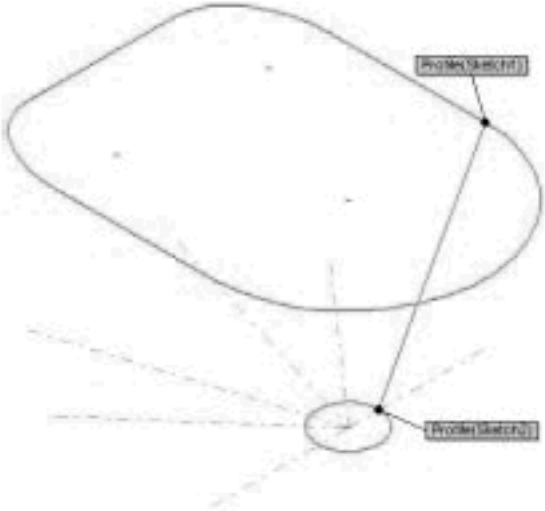
4 **Divide circle.**
Using the **Split Curve** command, add six split points, breaking the arc into pieces. Make each split point coincident with a centerline. You can add **Coincident** relations or you can drag and drop them onto the centerlines.



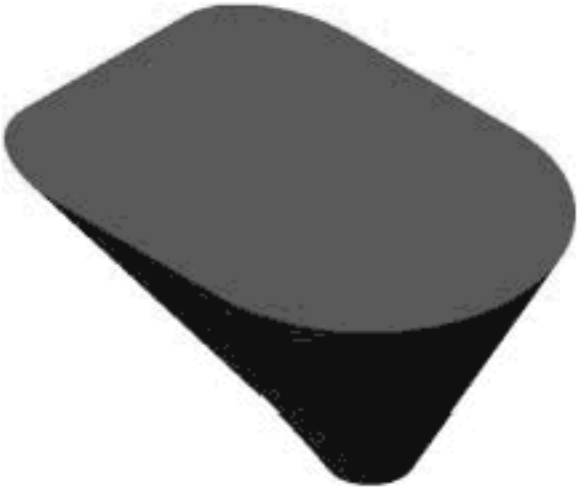
5 **First loft.**
Exit the sketch and loft between the two profiles. Select two endpoints that will match position, one from each sketch. This will ensure that the "start point" of the loft will be positioned correctly.

The option **Maintain Tangency** should be used.

An extra callout was added to the illustration for clarity.

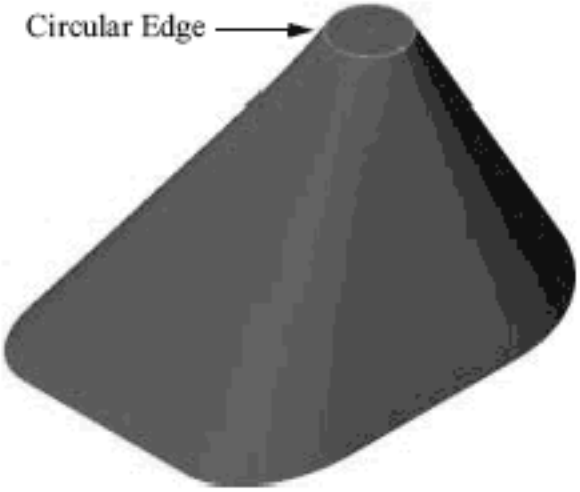


6 **Resulting loft.**
The loft solid should look like this when completed.

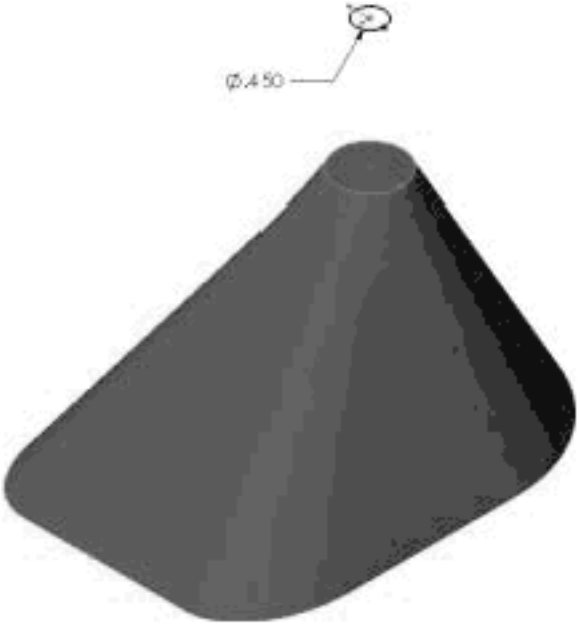


7 **Initial neck sketch.**
The funnel neck is formed by another loft, this time using two sketched circles. Flip the model over and sketch a circle on the end face, making it **Coradial** with the circular, outer edge.

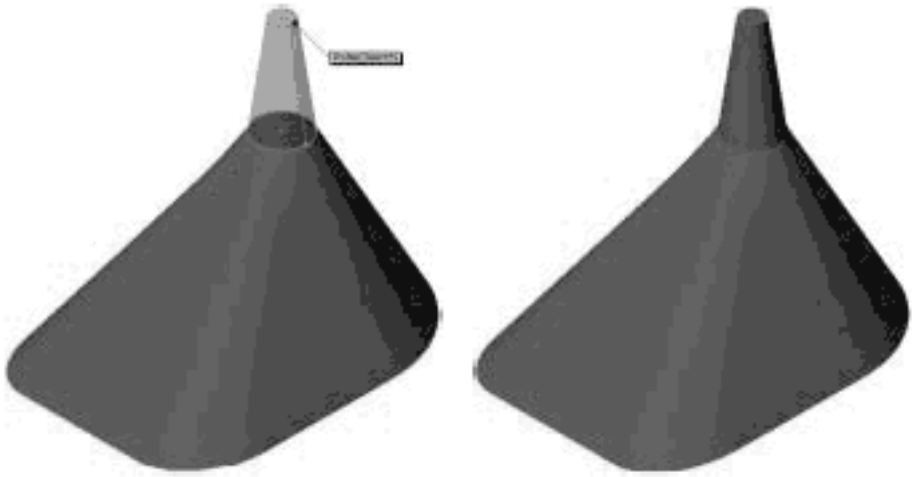
Add a point related to the `Origin` with a **Vertical** relation on the edge.



8 **Neck end sketch.**
Create a new reference plane offset from the circular face by **2"**. Sketch a circle that is lined up with the `Origin`. Add a point on the circumference of the circle that is related to the `Origin` with a **Vertical** relation. The points are used to "line up" the profiles just as the centerlines were used in the first loft.



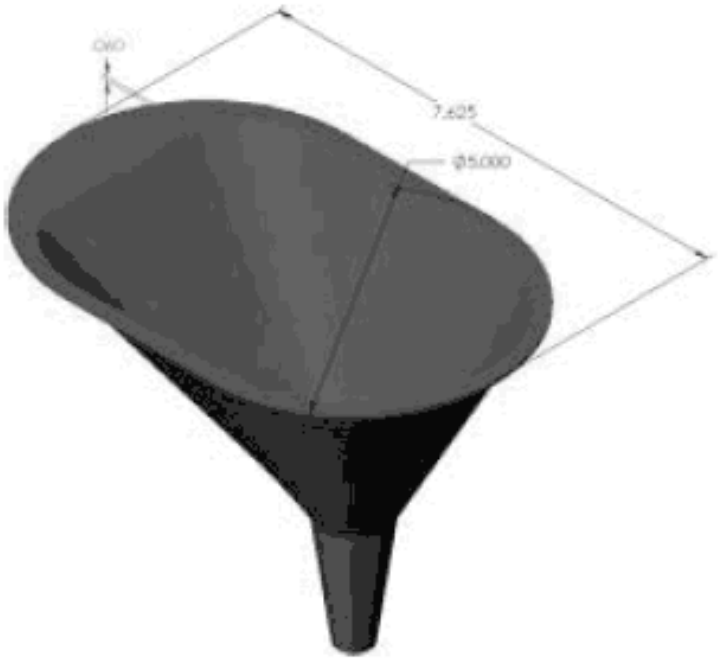
9 Neck loft.
Using the point entities to select the sketches, loft between the profiles.



10 Shell the funnel.
The dimensions are given for the *inside* of the funnel. Create a thin walled part by shelling to the *outside*, a thickness of **0.06"**.

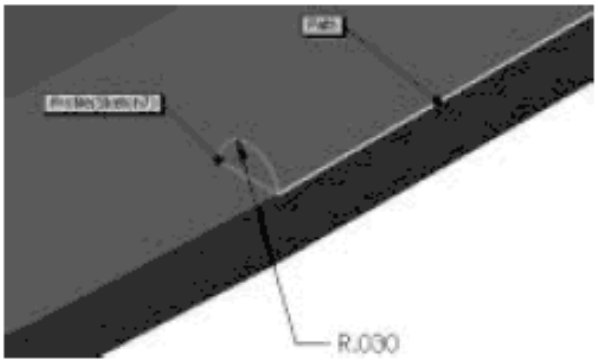
11 Build the rim.

Sketch the outline of the rim using the dimensions given. Use **Convert Entities** to create the inner outline. Extrude the rim to a depth of **0.06**". If desired, use **Link Values** to tie the two thickness values together.



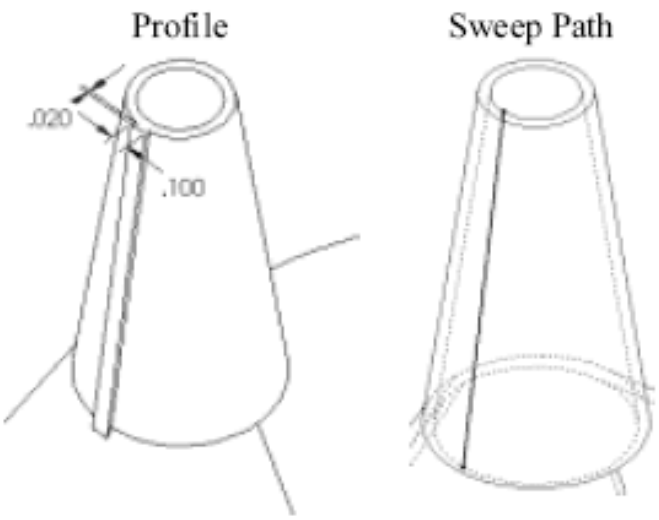
12 Sweep a lip on the underside of the rim.

The cross-section of the lip is a semi-circle, **0.060**" in diameter. Use the model edge of the rim as the sweep path.



13 Make a rib on the neck of the funnel.

Funnels don't work well if air can't get out of the bottle. Sweep the section along a curve that lies on the inner face of the funnel neck.



An easy way to construct this curve is to sketch a line and constrain it with **Pierce** relations to model edges at the opening and where the *inside* of the neck meets the main body.

14 Pattern the rib.

Make a total of three ribs, equally spaced, using a circular pattern.

15 A hole in the rim.

Using the dimensions provided, sketch a profile to cut through the rim so the funnel can be hung on a hook. Notice the use of an angular dimension on an arc. This can be created by picking the arc's centerpoint and its two ends.